Instructions to Authors for the Preparation of Papers for the 7th Asia-Pacific International Symposium on Aerospace Technology (APISAT)

Please select category below:

Normal Paper ☐ √

Undergraduate Student Paper ☐

Postgraduate Student Paper ☐

The research on the drag reduction of a transport aircraft with upswept afterbody using long fins

Abstract

The aerodynamic characteristics of the upswept tail of a transport aircraft are studied and different fin plans to reduce the drag are evaluated. Two symmetric vortices emerge under the fuselage afterbody, which are the major causal factor of the additional pressure drag. A pair of fins installed under the fuselage extruding the core of the vortices effectively damp the vortex. Parametric study shows that the length, height, location and yaw angle of the fins are the sensitive factors of drag reduction. Drag reduction of 21 counts is achieved in wind tunnel test for typical cruise angle of attack. The pressure recovery of the bottom surface of the tail is improved by adding the fins and the effect is higher for smaller angle of attack, which causes nose down pitching moment, reduced longitudinal stability, as well as the reduction of pressure drag.

Keywords: upswept tail, drag reduction, fin, afterbody, vortex flow, pressure recovery.

Introduction

The tails of conventional airliner or cargo aircraft all have upswept angles of certain degrees. For airliners the upswept tail minimizes the undercarriage height required by ground clearance, while for cargo or military aircraft there is an additional function of locating cargo gates under the tail, which allows airdropping during flight or dropping down as ramp on ground in hostile environments. There are 2 types of concepts about the tails with cargo door: the earlier transport aircrafts with turbo-propeller engines such as C-130, C-160 and AN-12 adopt large upswept angles to meet the requirement of cross section area for cargo with minimum cutout length, and the associated drag penalty is high. The aircrafts with turbofan engines such as C-141A, C-5A and Il-76 have smaller upswept angles, smoother tail shape and larger tail fineness ratio, which reduce the drag with the cost of longer cutout length[1].

The upswept tail generates additional drag and specific flow structure in surrounding flow domain. The drag increment can be estimated by cross flow theory, which assumes that the pressure drag can be calculated by the flow vector perpendicular to the fuselage with consideration of angle of attack (AOA) and the tail upswept angle. For circular cross section the drag coefficient based on cross flow dynamic pressure is about 1, while the drag coefficient for rectangular section with rounded corner reaches 1.5 to 2[2-3]. Researches show that the cross section shape, the variation of the cross section shape along the fuselage axis, upswept angle, fineness ratio and contraction ratio are the major parameters that affecting the pressure drag of the afterbody. When the AOA is less than the upswept angle the frictional drag is nearly constant when the AOA increases, but the pressure drag decreases [4-6]. The characteristics of flowfield around fuselage tail have been investigated by wind tunnel tests (using oil flow, smoke, laser, et al) and Computational Fluid Dynamics (CFD) by many researchers, and the

major conclusions are: cross flow emerges under the fuselage tail, and a pair of symmetric vortexes are observed in the space near the body and move downstream. The vortex system shifts from lower vortexes, none vortex to upper vortexes when the AOA change from negative to positive. Higher upswept angle enhances the afterbody cross flow and the separation of flow.

The reduction of the drag of fuselage afterbody has drawn wild attention around the world, as it usually accounts for 15% to 20% of the total drag of the fuselage, or 5% to 7% of total aircraft drag. Two methodologies are commonly adopted to reduce the afterbody drag. The first is the optimization of the tail parameter and shape, the second is the installation of flow control devices to change the flow structure. The first method suggests lower upswept angle, optimized cross section, larger fineness ratio, smaller contraction ratio et al to avoid or suppress vortex flow and reduce the drag, and is applicable to preliminary stage of fuselage design[4][5][12]. The second method installs flow control devices on the baseline configuration to reduce the drag, hence is applicable to wider situations. Wu installed a pair of chine on the tail at the starting point of vortex flow, which depressed the separation vortex and reduced the drag for 9 counts at cruise condition [13]. Yu and Du investigated the effects of vortex generators (VGs) at different location and incident angle on the control of afterbody separation, and reduced the drag of the aircraft by around 1% [14-15]. Wortman validated the effects of VGs on Boeing 747 and C-5A in wind tunnel tests and the drag reduction rates based on fuselage drag were 3% and 6% respectively [16]. Xia studied the effect of different flow control devices on aircraft afterbody, and the small plates reduced the fuselage drag by 10.8% at 0 degree of AOA, but the drag reduction dropped to 5.7% when the horizontal tail was installed[17].

The above mentioned researches reveal detailed information about the flow structure of the upswept tail and achieve significant progress in drag reduction of the tail. However, improvements are still needed for that most researches are based on single fuselage, with the ignorance of the complex interference with the other components of aircraft. The suggested flow control devices are generally small in size and the drag reduction rate is less than 10% of the drag of fuselage. This report studies the flow characteristics of the afterbody of a complete aircraft in production, and designs a pair of long fins to significantly damp the vortex flow, achieving much higher drag reduction rate, hence is highly attractive to engineering application.

General information of the research

A. The geometry of the upswept tail

The transport aircraft has flat bottom under its tail for cargo door, and the upswept angle is 11 degrees. The typical cross section of the tail consists of an upper arc, a straight line in the bottom, and the connecting arcs of small radius, as shown in figure 1. Based on the geometric properties, strong vortex flows are expected under the tail, and the pressure drag could be high. The suggested fin plans in the research are installed on the flat portion of the tail bottom and are convenient to be manufactured.

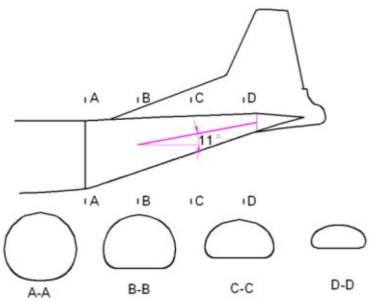


Fig.1 The sketch of the upswept tail of a transport aircraft

B. The method of numerical simulation and wind tunnel test

The effects of the fins on aircraft aerodynamic characteristics are studied by numerical simulations and wind tunnel tests. The numerical simulation is based on finite volume method, and solves Reynolds Averaged Navior-Stokes (RANS) equation to obtail the aerodynamic force and flowfield. Mixed unstructured mesh is adopted to discretize the flow domain, and mesh density is set higher in critical areas where vortex or separation flows exist. Prisms are created normal to wall boundaries according to the meshing strategy recommended in reference 18, and the y+ value is between 30 and 200. Wall function method is used to blend the variables between wall and outer flow, which increases the meshing efficiency with acceptable accuracy. The total mesh number for complete aircraft is around 15 million, and the sketch of mesh is shown in figure 2. The cases are iterated in commercial software FLUENT using pressure based coupled solver. The advection term is discretized by second order upwind scheme. The SST k-ω two equation turbulent model is adopted to model the viscosity effect, with turbulent energy k and dissipation rate ω discretized by upwind scheme.

The optimal fins selected by CFD are validated by wind tunnel test of a 0.07 scale model in the FL-12 tunnel of China Aerodynamics Research and Development Centre (CARDC). It is a single return low speed tunnel with a closed test section. The cross section is 4 metres in width and 3 metres in height. The wind speed can vary from 50 to 100 m/s, with turbulent degree less than 0.12%. In order to minimize the interference from supporting stings the test model is reversely installed, as shown in figure 2.

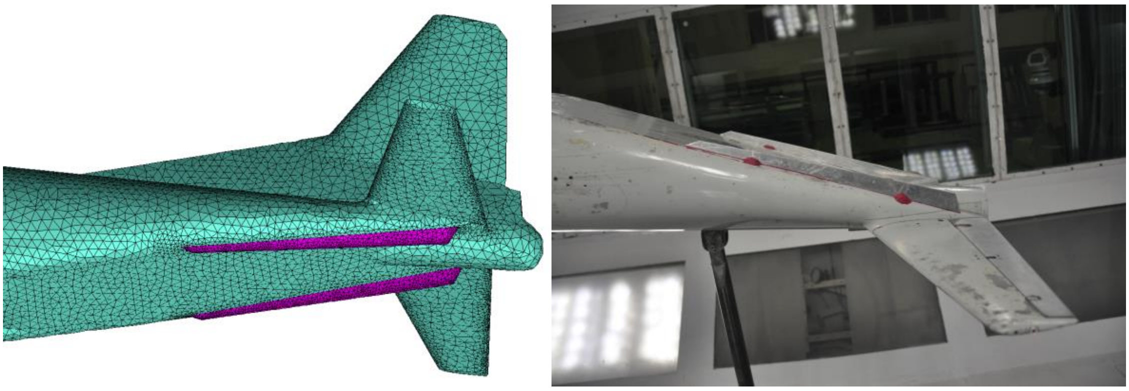


Fig.2 Sketch of the computing mesh and the test model

The flowfield characteristics of the upswept tail

The tail of large upswept angle and flat bottom promotes strong cross flow around it as shown figure 3. The cross flow velocity increases as the air moves downstream, and eventually forms a pair of vortex flow. The core of the vortex is close to the bottom surface of the tail, which makes it possible to install effective flow control devices on the fuselage. The existence of vortex flow structure creates low pressure areas on nearby fuselage, hence increases the pressure drag.

The vorticity which identifies the core of vortex flow demonstrates that the intensity of vortex flow is reliable on AOA, as shown in figure 4. As the AOA drops, the width of the vortex core increases, extending longer and lower downstream. The enhanced vortex flow leads to larger energy dissipation and higher pressure drag. The drag of fuselage part at different AOAs shows the same trend that higher drag is accompanied by lower AOAs, as shown in figure 5. It can be seen that when α is less than 8 degrees the drag increases rapidly. As the vicious drag is proportional to the wetted area which is constant in the case, it can be concluded that the drag increment is from the contribution of pressure drag. For typical cruise AOA=4 degrees the drag is higher than the minimum drag of the fuselage by 23 counts, which suggests that the potential of drag reduction is rather high.

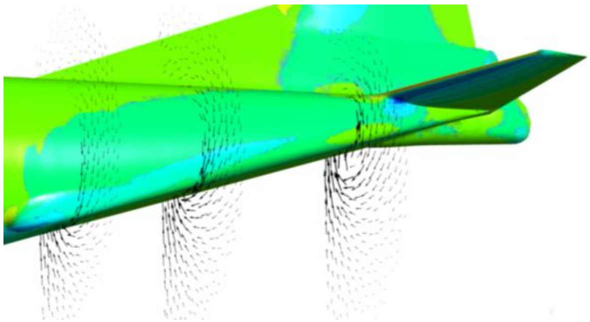


Fig.3 Projected velocity vector in cross section of aircraft tail (cruise angle, half model)

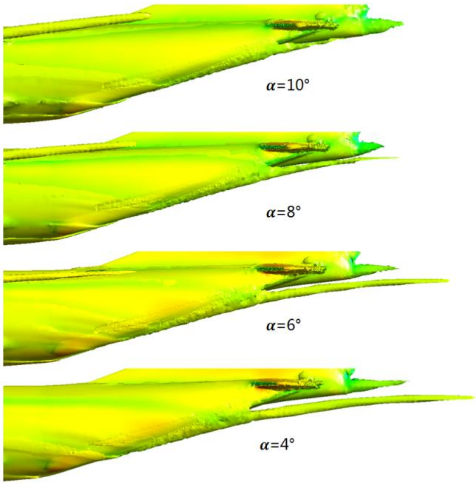


Fig.4 Vorticity of equal value around fuselage tail

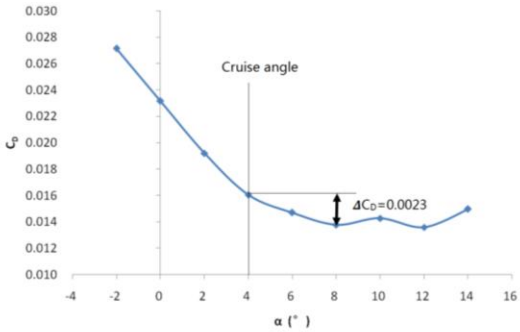


Fig.5 Drag characteristics of the fuselage (wind tunnel test)

Parametric study of the fins

The geometric parameters including length, height, installation position and yaw angle are studied to find the sensitive parameters, and the arrangement of the fins has to fulfill the constraints of structure at the same time. The selected parameters are varied systematically and evaluated in CFD to find the sensitivity on drag reduction, and dozens of fin plans are studied to find the optimum. The fins have 3 different sizes as shown in figure 6, and the lengths are 2.5m, 4.5m and 6.0m. The 2.5m fins are installed after the cargo door and are constrained in length by structure limit. The 4.5m and 6.0m fins are installed partly on the cargo door and partly on the fuselage, and need to be separated to 2 sections for cargo gate operation. Part of the fin plans are shown in figure 9. All the fin plans are rounded flat plate with thickness of 50mm for real aircraft.

The effect of the spanwise position on drag reduction is studied based on 4.5m fins, and the results can be seen in figure 7. The best value for spanwise location Z is 1.5m, where the fins intersect with the vortex core at the upstream of vortex and almost coincide with the vortex core. If the fins move to a Z value either smaller or larger the damping effect of them is reduced hence the drag reduction is less. Different heights of the fins significantly affect the effect of drag reduction, as shown in figure 8. When the height is higher than 0.5m the drag reduction decreases in linear relation with it. It can be seen that when the height increases secondary vortex is created by

the fin itself, which generates extra energy lost and higher drag. The height of the fins should be less than 500mm to achieve reasonable drag reduction. Typical fin plans and the corresponding drag reduction at cruise AOA are shown in figure 9. The fin plans with no yaw angles have poor performance in drag as they are not aligned with the direction of the vortex core. The yawed fins achieve certain degree of drag reduction, but the effect is limited. The longest 6.0m fins show the best effect in drag reduction, and the drag reduction reaches 0.0028 when properly arranged.

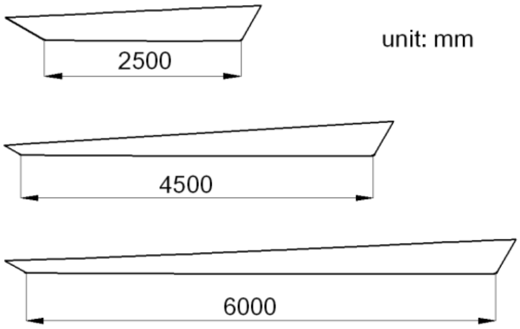
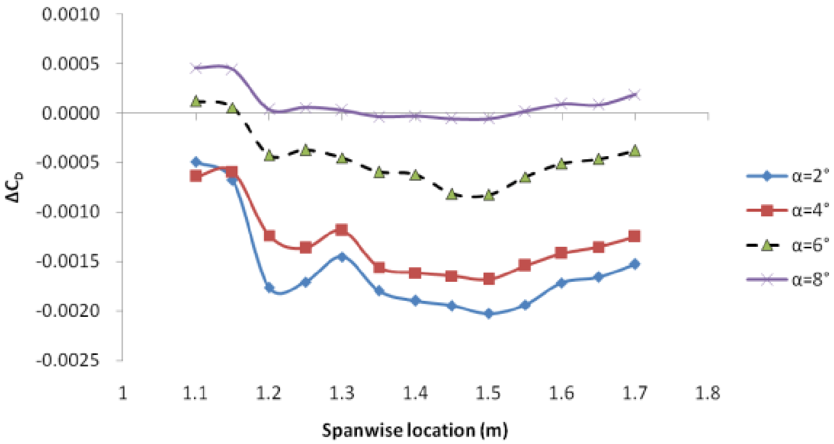
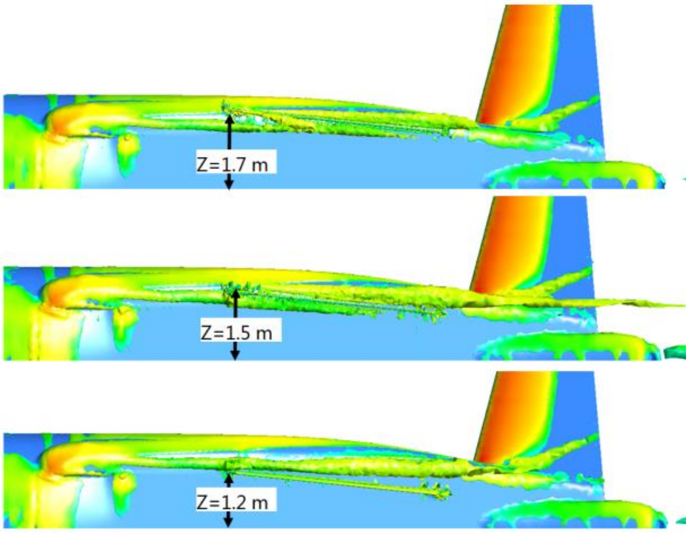


Fig.6 Geometry of the drag reduction fins

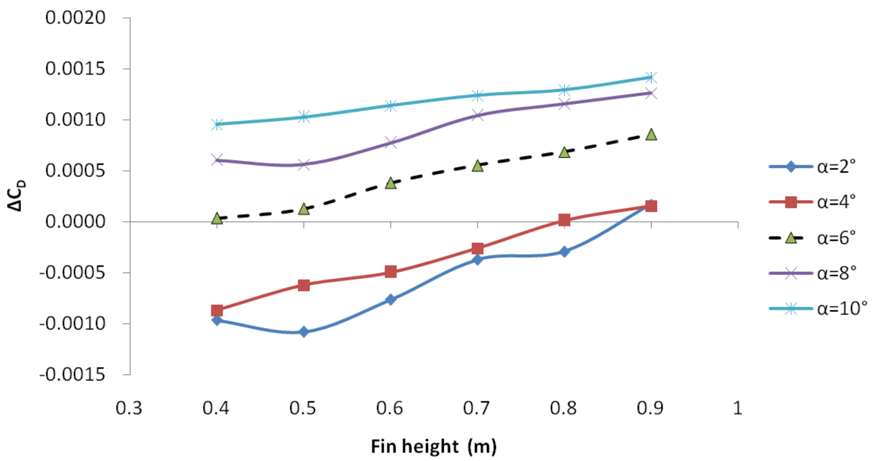


(a) Drag reduction effect of fins at different spanwise location

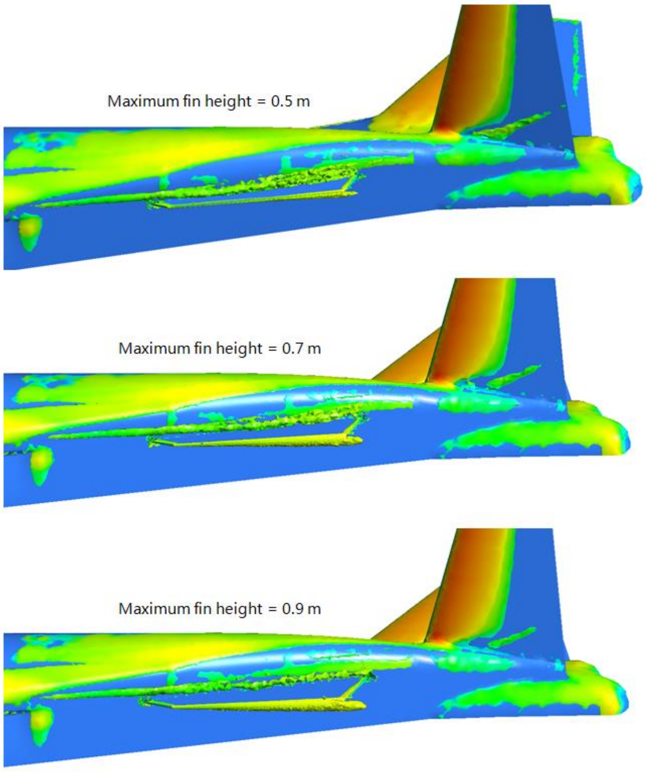


(b) Flow vorticity with fins at different spanwise location

Fig.7 Drag reduction and vorticity with fins at different spanwise location



(a) Drag reduction effect of fins of different heights



(b) Flow vorticity with fins of different heights

Fig.8 Drag reduction and vorticity of fins of different heights

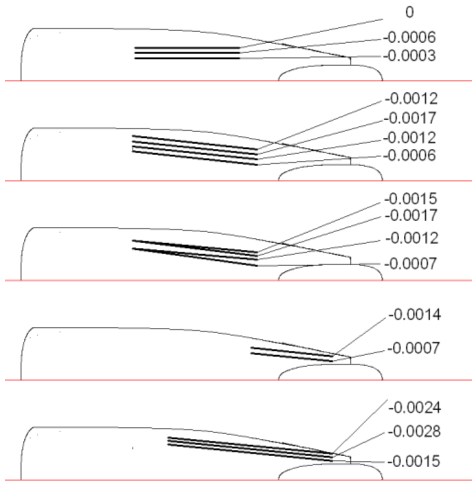


Fig.9 Drag reduction of different fins (α=4°)

The aerodynamic characteristics of the optimal fins

The optimum fins selected by CFD are tested in wind tunnels, and the aerodynamic characteristics of them are shown in figure 10 to 13. The picture of the vorticity shows that the fins effectively depress the strong vortex flow of the baseline aircraft. The large scale core of the vortex flow can no longer be seen with fins installed, indicating that the energy dissipation is reduced significantly. The drag reduction from wind tunnel test is 21 counts at cruise AOA and has same trend with CFD, which shows that the CFD is fairly valid in simulating this type of flow. The discrepancy between the test and CFD is possibly caused by the modeling capacity of the numerical simulations, because the fidelity of the unstructured mesh is not high enough, and the interference between the vortex and the fin might be too complex to be fully predicted by RANS models.

The fins also affect the pitching moment of the aircraft, showing reduced longitudinal stability and nose down tendency. The major reason is that the fins recover the pressure of the tail, generating positive lift and nose-down pitching moment. This effect decreases as the AOA is larger, which creates negative lift slope on the tail and reduces the longitudinal stability. The comparison of the pressure distribution before and after the installation of the fin from CFD shows the trend of the increased pressure on the bottom of the tail.

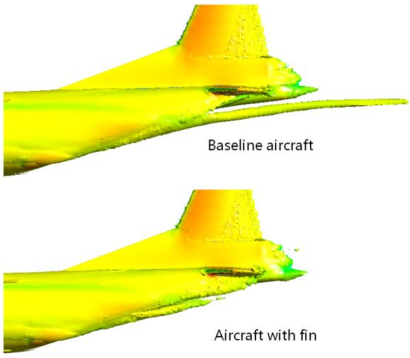


Fig.10 The effect of fins on flow vorticity(α=4°)

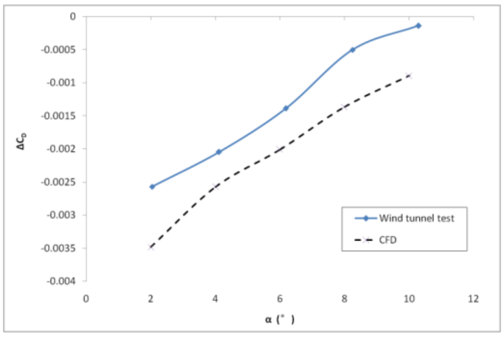


Fig.11 The quantity of drag reduction from CFD and wind tunnel test

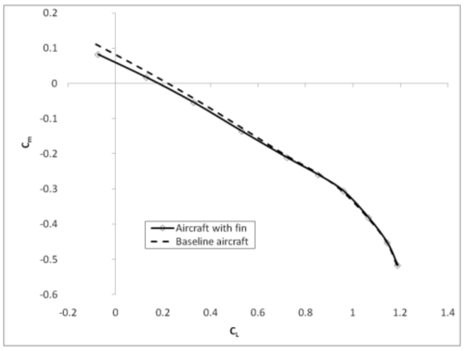


Fig.12 The effect of fins on pitching moment (test)

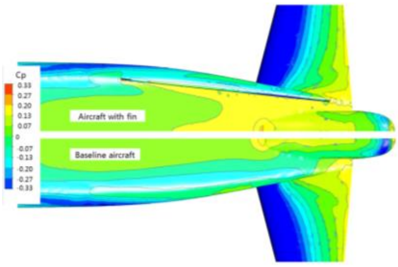


Fig.13 The effect of fins on pressure distribution of fuselage bottom (CFD)

Conclusion

The flow structure of the upswept tail is studied by CFD, and the symmetric vortex flow is found to be the major causal factor of the high pressure drag. Fin plans are designed parametrically to suppress or eliminate the vortex flow, and the parameters of the optimal fins such as length, height, position et al are selected. The wind tunnel test demonstrates that the optimal fins reduce the drag by 21 drag counts at cruise AOA, which can significantly improve the performance of the aircraft. The CFD method achieves similar results of drag reduction compared with tests, and is capable to obtain the detailed flowfield, hence is effective to investigate the aerodynamic characteristics of the upswept tail.

References

1. Committee of <Manual of aircraft design>, *Manual of aircraft design Vol. 6 aerodynamic design*, Aviation Industry Press, Beijing, 2002, pp.121. (in Chinese)

2. Raymer, D.P., A*ircraft design: a conceptual approach,* 4th edition, AIAA Education Series, Reston, VA, USA, 2006, pp. 333-335.

3. Torenbeek, E. *Synthesis of subsonic airplane design*, Delft University Press, Delft, Holland, 1982, pp. 504-505.

4. Zhang, B.Q., Wang, Y.Y., and Duan, Z.Y., et al. "Design method for large upswept afterbody of transport aircraft". *Acta Aeronoutica et Astronautica Sinica*, Vol. 31, No. 11, 2010, pp. 1933-1939. (in Chinese)

5. Mao, X.M., Zhang, B.Q., and Wang, Y.Y., "Research on parameters influence of fuselage afterbody with large upswept angle of transport aircraft", *Aeronautical Computing Technique*, Vol. 41, No. 1, 2011, pp. 78-81. (in Chinese)

6. Kong, F.M., Hua, J., and Feng, Y.N., et al. "Effects of geometry parameters and flow parameters on drag coefficient of upswept afterbodies", *Journal of Beijing University of Aeronautics and Astronautics*, Vol. 29, No. 1, 2003, pp. 39-42.(in Chinese)

7. Kong, F.M., Hua, J., and Feng, Y.N., et al. "Investigation of the flow mechanism on the afterbody with larger upswept angle", *Acta Aerodynamica Sinica*, Vol. 20, No. 3, 2002, pp. 326-330.(in Chinese)

8. Huang, T., Deng, X.Y., and Wang, Y.K., et al. "An experimental study of flow patterns over the upswept after-body of a civil aircraft", *Acta Aerodynamica Sinica*, Vol. 21, No. 1, 2003, pp. 29-37.(in Chinese)

9. Kong, F.M., Hua, J., and Deng, X.Y., et al. "Analysis of the flows and the drag about upswept afterbodies", *Acta Aerodynamica Sinica*, Vol. 21, No. 1, 2003, pp. 67-74.(in Chinese)

10. Zheng, L.Q., and Yang, Y., "Influence of landing gear cabin on drag of a transport aircraft body", *Aeronautical Computing Technique*, Vol. 40, No. 4, 2010, pp. 56-58. (in Chinese)

11. Wang, J., *Aerotransport profile design based on UG secondly development and analysis of the aftbody's flow distribution*, Nanjing University of Aeronautics and Astronautics, 2009. (in Chinese)

12. Shan, D., and Wang, Z.P., "Research on aerodynamic characteristics of rear of transport aircrafts", *Aeronautical Computing Technique*, Vol. 43, No. 4, 2013, pp. 79-82. (in Chinese)

13. Wu, N., Duan, Z.Y., and Liao, Z.R., et al, "Research on chine of aerotransport after-body for drag reduction and stability enhancement", *Acta Aerodynamica Sinica*, Vol. 30, No. 2, 2012, pp. 223-227.(in Chinese)

14. Yu, Y.Z., Liu, J.F., and Jiang, Z.Y., et al. "The investigation of flow control and drag reduction mechanism for transport airplane aft-body", *Acta Aerodynamica Sinica*, Vol. 29, No. 5, 2011, pp. 640-644.(in Chinese)

15. Du, X.Q., Jiang, Z.Y., and Tong, S.X., et al. "Experimental study on control of separated flow over aft-body and drag reduction mechanism by using vortex generator", *Engineering Mechanics*, Vol. 29, No. 8, 2012, pp. 360-365.

16. Wortman, A., "Reduction of fuselage form drag by vortex flows", *Journal of Aircraft*, Vol. 36, No. 3, 1999, pp. 501-506.

17. Xia, X.J., and Ma, S.L., "The effect of afterbody devices on drag reduction", *Acta Aeronoutica et Astronautica Sinica*, Vol. 12, No. 8, 1991, pp. 435-438. (in Chinese)

18. ANSYS, *ANSYS FLUENT theory guide* [electronic resource], ANSYS, 2011, pp. 119-135.